A Parametric Template Format for Solid Models of Reinforced Concrete Structures

Guillermo A. Riveros[1], Kerry T. Slattery[2]

[1]U.S. Army Engineer Research and Development Center, Vicksburg, MS, USA, 39180

[2]Missouri State University, Springfield, MO, USA, 65897

[1]Guillermo.A.Riveros@usace.army.mil, [2]KerrySlattery@MissouriState.edu

Abstract

The trend in the architecture/engineering/construction industry toward the use of three-dimensional representations of structures in design, analysis, and construction has led to the acceptance of the CIS/2 format for steel structures. This allows all information defining the structural steel to be passed among the structural designer, detailer, fabricator, and erector through a digital data file that eliminates the need for re-entering information. The three-dimensional model also facilitates the task of checking for interferences and inconsistencies.

A format is proposed for describing the geometry of typical reinforced concrete structures as a function of user-defined parameters. The structure is constructed from components aligned with a flexible grid; corners are analogous to finite element nodes, while beams and slabs would be one- and two-dimensional elements, respectively, in conventional structural models. The dimensions of the components are defined as functions of parameters listed in the template. A simple application of this approach would be a template to model a multiple-story concrete frame structure, and the basic template can be enhanced to give the designer more flexibility throughout the structure by adding parameters.

Two applications of the template format are developed: a simple reinforced concrete frame and a more complex reinforced concrete pumping station. The graphical user interface allows the designer to change any dimensional parameter to immediately update the structure geometry in order to meet the project requirements. The final solid model is then converted to a finite element model and analyzed to determine shear, moment, and axial force in beams, columns, and slabs, which can then be used to design the reinforced concrete members. A sampling of the results of the analyses is presented.

Introduction

The finite element method as applied to structures has evolved with advances in the capability and economy of computers. While researchers have frequently used the additional power to study the nonlinear response of detailed models, the typical engineering designer is satisfied with linear analysis using simple beam and shell elements. The engineer is interested in improving the efficiency and quality of the design process. This requires the development of sophisticated pre- and postprocessors that minimize the time required to describe the structure and the effort required to convert results to reliable designs. General purpose preprocessors such as Patran® introduced in the early 1980s [1] began with text input complemented with graphical feedback and output. Even with the development of event-driven graphical user interfaces for input in the late 1980s, these programs still required a trained and experienced engineer to run them efficiently, and the output required further manipulation to produce an engineering design. Current commercial software allows finite element models of simple frame structures to be quickly generated given column spacing and story heights [4]. Efforts to automate the design of structural elements have also been implemented, and software is available to automatically detail connections in steel frames and export the information to numerically controlled manufacturing equipment. However, any deviation from standard layouts requires more user skills or additional program development to include specialized details.

Software to design complete reinforced concrete structures is less capable. The challenges include the semi-empirical basis of many concrete design requirements, the complexity of designing reinforcing bar embedment,

and the absence of standard shapes for reinforced concrete. The Portland Cement Association markets programs to design structural components such as beams, columns, and slabs, but these primarily perform code checks and aren't designed to model, analyze, and design an entire structure. Applications are also available to quickly design simple structures such as box culverts, retaining walls, and precast sections [2, 3]. The Corps of Engineers, through the Computer-Aided Structural Engineering (CASE) committee, has assembled a library of computer programs to assist in the design of reinforced concrete navigation structures [4, 5, 6, 7, 8]. The next generation of reinforced concrete design software should permit the engineer to easily describe the structure and input, and analyze required load cases, output the design for the structural members, and modify designs based on these results. The structural model should be plotted to scale in three dimensions so it can be easily and accurately transported to Computer Aided Drafting (CAD) software for detailing. Future generations of programs should include computerized detailing and be able to produce output in a format to assist in construction including, for example, automation of estimating and rebar fabrication.

An approach for modeling concrete structures is proposed that can greatly improve design efficiency through the development of parametric models and enhance interoperability by describing the actual three-dimensional geometry in a format able to be standardized. Once a template is developed for a structural configuration, the designer simply changes the values of parameters defining the dimensions for the new structure and updates load magnitudes and positions. The information is stored using a standard format that can be read and utilized by other software applications. The proposed format describes the information required for a solid model of a reinforced concrete structure. It is currently displayed in a simple text file that can be easily converted to an XML schema and adapted to a standard Building Information Model format such as the IFC/ifcXML Common Model [9].

Model Assumptions

Any standard format to describe a structure will have some limitations. It is not possible to provide for any possible geometry. The goal of this development is to provide enough flexibility to accurately model the majority of concrete structures and to allow for future enhancements that can broaden the capability. Structures are modeled as an assemblage of slab, beam, and corner components as shown in Fig. 1 for a hollow concrete cube. The individual components are shrunk in the plot for clarity.

- *Substructures:* A model can be composed of any number of substructures that are assumed to be connected where exterior surfaces come in contact.

- *Layout Grid:* Each substructure must be aligned with a three-dimensional Layout Grid as shown in Fig. 2 for a two-story building. Structural components are identified by their location in the grid. The default grid is orthogonal. Individual grid lines may be offset, causing intersecting lines to be skewed from the orthogonal grid.

- *Corners:* Corners are hexahedrons located at the intersection of grid lines.

- *Beams:* Beams are aligned with grid lines and may represent actual beams (horizontal spanning members), columns, or parts of continuous slabs.
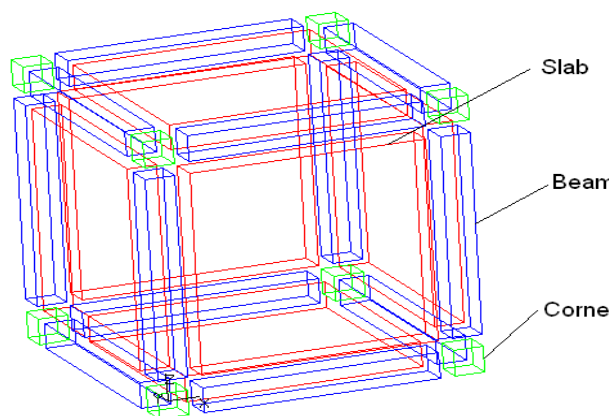

FIG. 1. A HOLLOW BOX MODELED USING SLAB, BEAM, AND CORNER COMPONENTS
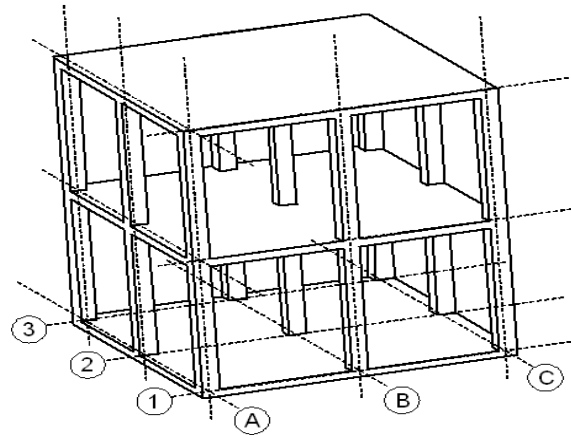
FIG. 2. A 3X3X3 LAYOUT GRID FOR A SIMPLE FRAME STRUCTURE

- *Slabs:* Slabs may be a floor slab or wall. They fill the space between four grid lines. A slab is defined as existing or not. Dimensions are still required for nonexistent slabs to define the spacing and thickness of adjacent beams. Each slab may have one hole pattern.

- *Footings:* Either wall or column footings may be specified. These are centered on the corresponding wall or column. They are treated as slabs but can be placed only on the bottom plane. They connect to the vertical slab and beam elements at the bottom of the structure.

Template Format

A Microsoft Excel spreadsheet is used to organize the information required to describe the reinforced concrete structure. A Visual Basic for Applications program is then used to write the template to a text file. The following discussion describes the information provided in the template.

Each component in the model is identified by a series of numbers: S, I, J, K, M, where S is the substructure number; I, J, and K are the least; X-, Y-, and Z-direction gridline numbers intersect at one node of the component; and M has a value of 0, 1, or 2 indicating X, Y or Z, respectively. The value gives the direction of the normal to a slab or the direction of the axis of a beam. A value for M is not required for corner components.

Many quantities in the template are expressed as functions of parameter values. The functions must be expressed as linear combinations of parameters, for example, 0.5*P3-1.5*P8+P12.

Zones and elastic foundations are defined for slabs. The definition must also indicate which of the adjacent beams and corners share that property. Two four-bit integer values from 0 to 15 are used that, when converted to binary, tell which of the four adjacent components are included. Conventions for these affected beams and corners along with face numbers and length, width, and thickness orientations follow a system consistent with typical finite element numbering schemes.

Model Characteristics

The following 20 parameters must be provided to define the model:
- *ElSlabLong:* the number of superelements used along the length of beam components and in both in-plane directions in slab components in the finite element model. A value of 3 is adequate for developing the template, but greater refinement is required for sufficient accuracy in the finite element analysis.
- *Footings:* a Boolean that is true if footings are to be modeled
- *Wall:* a Boolean that is true if there are basement walls with wall footings
- *NumParam:* the number of parameters (dimensional and load) used in the model
- *Enhanced?:* a Boolean that is true if the file includes information to tie the parameter values to geometric entities in the model for display
- *numSubStruct:* the number of substructures in the model
- *GridX, GridY, GridZ:* the number of gridlines in each direction for each substructure

- *numOffsets:* the number of beam components that are offset from the orthogonal grid
- *numLoadCombs:* the number of load combinations computed in the structural analysis
- *numLoadCases:* the number of load cases in the structural model
- *numZones:* the number of zones, which are groups of slab faces that will be assigned a common area load; for example, a fluid pressure
- *numProfiles:* the number of load distribution profiles defined for the model
- *numPointLoads:* the number of concentrated loads
- *numLineLoads:* the number of loads distributed along a line
- *numAreaLoads:* the number of loads distributed over a surface
- *numHoleLoads:* the number of loads applied to the surfaces around the hole
- *numVolLoads:* the number of loads applied to the total volume. The gravity load case is a volume load that is automatically defined as Load Case 0.
- *numMats:* the number of material properties

Parameters

Parameters may be used to define linear dimensions in the structure or force magnitudes. Functions of various parameters are computed when generating the model and assigning loads. Parameters are named Pn where n is an integer. For example, P1, P2, and P3 would be the first three parameters in any model. An example parameter definition would be:

P1, EWSpace, 360, XY, X, 1

Where P1 refers to the first parameter, EWSpace is the parameter name, 360 is the default value, XY indicates that the parameter value will be displayed in the XY view of the model, X is the local direction of that dimension in the display plane, and 1 is the number of spaces the parameter will be displayed from the model. Three views are available for dimensional parameters: XY, YZ, and XZ. The view for parameters used to define loads are designated as L*aan* where *aa* is one of the three views and *n* is the load case number, for example, LXZ2.

Offsets

Offsets define the distance a beam component is offset from the orthogonal grid. The template entry gives the five-number beam designation, the offset direction (X, Y, or Z), and a parametric equation for the offset magnitude.

Profiles

Pressure profiles are used to define pressure distributions on surfaces. The pressure must be a function of X, Y, or Z and piece-wise linear. The first line of input for each profile gives the number of points n (n ≥ 2) and the independent variable X, Y, Z, -X, -Y, or -Z. The n subsequent lines are pairs of values giving the coordinate value and the magnitude of the pressure.

Load Combinations

Each load combination is defined in one line as a series of multiples for each load case contributing to that combination.

Point Loads

Concentrated load definitions give the load case number, the position of the load, and the magnitude of the load in each of three directions. All entries except the load case are defined as parametric equations. An example point load definition is:

4, P66, P67, P13+P14+P15, P57, P57, P60

The load is assigned to Load Case 4 applied at (X, Y, Z) = (P66, P67, P13+P14+P15) with a magnitude of P60 is the Z direction. P57 was defined to be zero so there is no load in the X or Y direction. This was a wheel load from a vehicle parked on the structure.

Line Loads

The definition for line loads is similar to point loads with the addition of three parametric equations so that the coordinates of the two endpoints of the line are defined. The load magnitude is defined in force/unit length.

Area Loads

Area load definitions are simplified by the use of Zones and Pressure Profiles. The required entries are Load Case number, Zone number, Pressure Profile number, and a parametric equation for the Datum. The Datum is the coordinate value that corresponds with zero in the pressure profile definition.

Hole Loads

The hole load option allows pressure loads to be applied to the surface surrounding a hole in a slab component. The load case number and five-number slab designation must be supplied, along with the Pressure Profile number and the four-bit affected Edges number.

Volume Loads

The gravity volume load is automatically defined as Load Case 0. Additional volume loads are defined by giving the direction as X, Y, Z, –X, -Y, or -Z and the volume multiplier in units of force/volume.

Material Properties

Standard properties for concrete and reinforcing steel are defined. The default concrete is Material 1 and reinforcing steel is Material 2. Additional concrete and steel materials may be included.

Interfaces

Interfaces are contact planes between adjacent substructures. The definition requires the two substructure numbers and four sets of parametric equations defining the coordinates of the corners of the interface plane. No attempt is made to align the finite element meshes in the substructures. The current finite element application generates additional nodes over a uniform, two-dimensional grid to simulate dowel bars between the substructures. Additional finite element analysis development is required to provide the user with better modeling options that will be facilitated by this information.

Substructures

The slabs in each substructure are defined in detail. The first entries for each substructure are the number of slabs with holes and the number of slabs on elastic foundations in the substructure. The next line gives parametric equations to define the (X, Y, Z) coordinates of the substructure origin. These are followed by six labeled sections for each substructure: X Slabs, Y Slabs, Z Slabs, Holes, Elastic Foundation, and Zones.

Slabs

Each slab is listed by giving its four-number designation (the substructure number is known), a Boolean value for whether or not the slab exists, parametric equations to define the thickness, length, and width of the slab, concrete and steel material property numbers, and an optional name. For clarity, slabs are assembled in groups of X Slabs, Y Slabs, and Z Slabs indicating the direction of the normal to the plane. An example of a slab definition is:

X Slabs,

0,0,0,0,True,P1-P2,P10+0.5*P11,P29+3*P11,1,2,SouthWall

Holes

Each slab with a hole is defined by the four-number slab designation, the hole pattern type, and up to four parametric equations indicating the location and size of the holes. The available hole patterns are: Circle, Rectangle, Double Circle, and Double Square.

Elastic Foundation

A list of slabs resting on an elastic foundation are defined using their four-number designation, the number of the face resting on the foundation, the foundation stiffness, the affected beams number, and the affected corners number.

Zones

The number of slabs in the substructure in each zone is listed and followed by a definition of each slab face, which includes: Face Number, four-number slab designation, affected beams number, and affected corners number.

Finite Element Approach

Solid finite element models of large structures can require a prohibitive number of degrees of freedom. The extensive use of superelements [10, 11] is expected in finite element applications using these templates. Slabs are currently modeled using 18-node superelements derived from eight, 8-node hexahedrons. These are similar to 9-node shell elements with the five- or six-degree-of-freedom shell nodes replaced by two, three-degree-of-freedom nodes at the top and bottom of the element. Methods to quickly synthesize entire slabs into single superelements that can accurately model complex configurations such as waffle slabs are under development. These can be easily integrated into the proposed template.

Example Applications

A finite element code is being developed along with the template to test and demonstrate new models. A new template is initially "enhanced" using the graphical interface to tie parameter definitions to geometric entities in the model. The program also provides postprocessing algorithms to display results. Models from two templates are presented to demonstrate the capabilities of the proposed format.

Reinforced Concrete Frame

Fig. 3 displays the side view generated from a template for a two-story, three-by-three-bay reinforced concrete frame. The four parameters shown are: Story Height, Floor Thickness, Footing Depth, and Footing Thickness. This template was developed with a minimal number of parameters so both stories have the same height, the floors and roof have the same thickness, and all footings are identical. Parameters displayed in the top view define the bay sizes and column and footing dimensions. The graphical user interface allows the designer to change any parameter and immediately update the geometry.

After entering the appropriate values for all dimensions and loads, the finite element model is automatically meshed and analyzed. The elements displayed in Fig. 4 are solid superelements derived from eight-node hexahedrons. This finite element approach is being developed to insure the integrity of the solid model.

Results displayed in Fig. 5 are moments in the slabs for a load combination that includes dead load and a lateral wind load. Design algorithms can use these results to determine the required reinforcing.
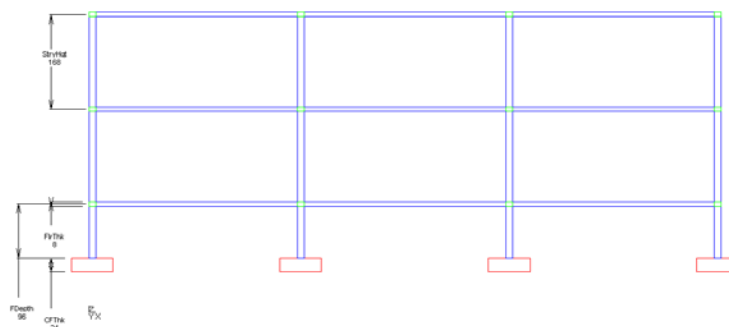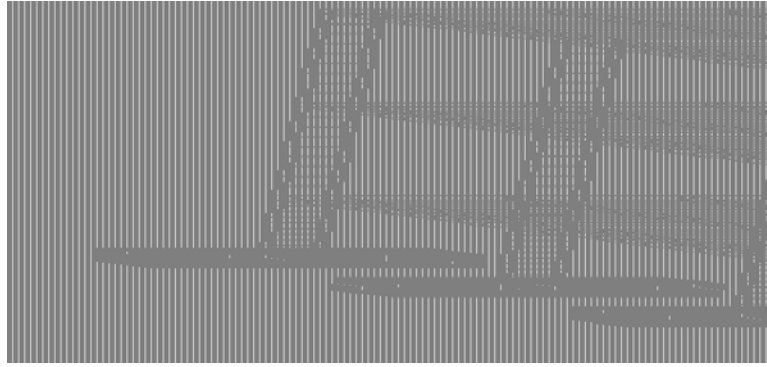

FIG. 3 GRAPHICAL DISPLAY OF TEMPLATE PARAMETERS

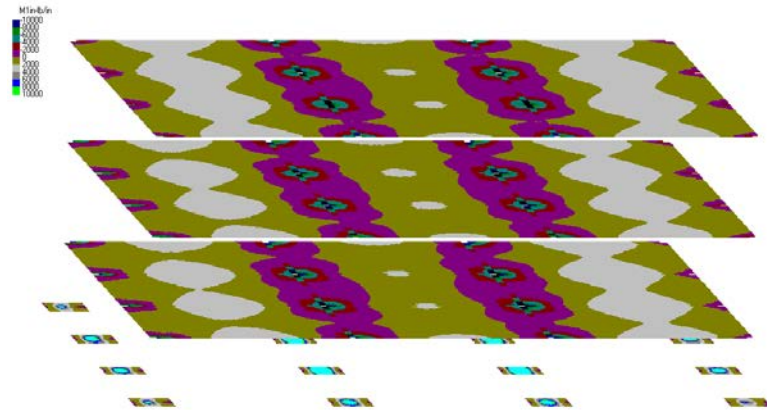FIG. 4 FINITE ELEMENT MODEL COMPOSED OF SOLID SUPERELEMENTS

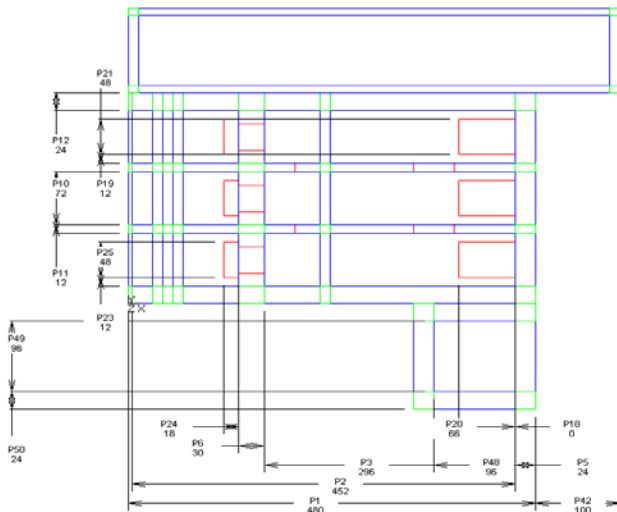FIG. 5 MOMENTS IN FOOTING, FLOOR, AND ROOF SLABS
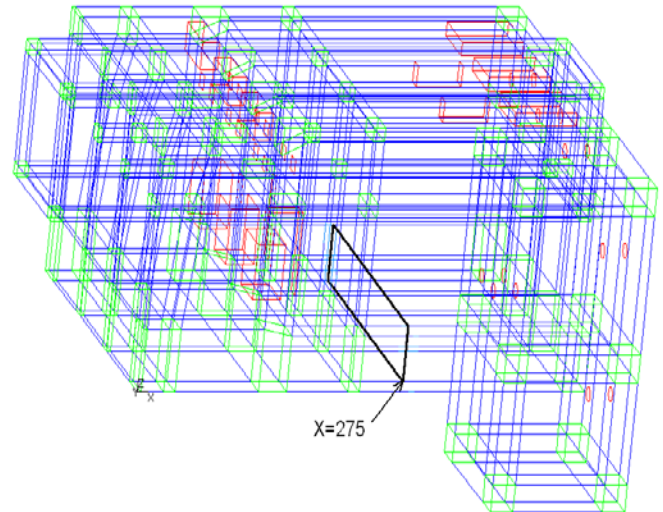

FIG. 6 TOP VIEW OF PUMPING STATION TEMPLATE

FIG. 7 ISOMETRIC VIEW OF PUMPING STATION TEMPLATE

Reinforced Concrete Pumping Station

While commercially available finite element codes address the design of the simple building described in the first example, the proposed format allows the development of templates for more complex concrete structures. The top view of a template for a pumping station is displayed in Fig. 6. The model includes the three-pump station along with attached culvert and sump structures. The model requires more than 50 parameters to define slab thicknesses, hole locations and dimensions, and the attached structures. Several of the parameters are displayed in the top view. They are simply labeled using the parameter number with the current parameter value shown in inches. A transparent, isometric view of the model is provided in Fig. 7. The positions of various round and rectangular holes defined in the model are more clearly displayed in this view.

Parameters for load cases defined in the template can also be modified through the graphical user interface. Fig. 8

shows the hydrostatic pressure load from water in one chamber of the pumping station. The depth parameter can be modified to reflect actual design conditions.

This model was also automatically meshed and analyzed under various load combinations. Fig. 9 displays Shear, Moment, and Thrust diagrams derived from the finite element results for the bottom slab along a cross section in the Y-Z plane at X = 275 inches. This slab cross section is highlighted in Fig. 7. The results are for the dead load case.
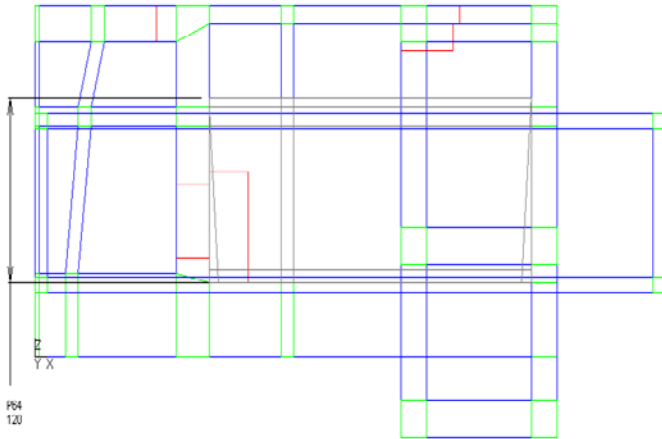

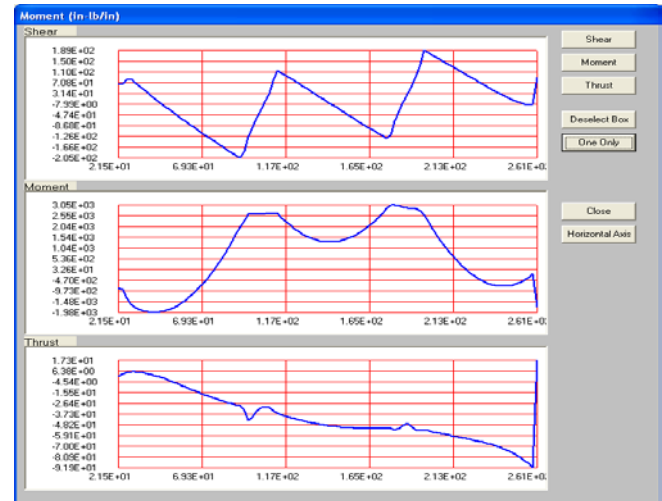FIG. 8 PRESSURE DUE TO WATER IN ONE CHAMBER IS DEFINED BY THE DEPTH (P64)

FIG. 9 SHEAR, MOMENT, AND THRUST DIAGRAMS

Conclusions

A template format for defining solid models of reinforced concrete structures has been developed. It can easily model simple concrete frame structures, but has also been shown to effectively model more complex structural geometries. The format is sufficiently flexible to add additional functionality, as required. The next step in the development is the creation of tools to assist with template design and the incorporation of more complex superelements to permit modeling and analysis of beam-slab and waffle-slab structures. The reinforced concrete frame example demonstrates the seamless process to generates the three story building template which uses a minimal number of parameters so both stories have the same height, the floors and roof have the same thickness, and all footings are identical. The results also provides the required moments in footing, floor and roof slabs required for the design calculations. Finally, the reinforced concrete pumping station example shows that the proposed methodology also allows for the development of more complex concrete structures. The model required more than 50 parameters to define the slab thicknesses, holes locations and dimensions, and the attached structures. Additionally, the parameters for load cases defined in the template are also prescribed which allows the design shear, moment, and thrust diagrams to be derived from the finite element results. These forces can directly be implemented in the design equations to verify the limit states capsulations.

REFERENCES

[1]   Weinberg, D. and T. Abbey. 2004. *An Historical Survey, Contemporary Assessment and Recommended Directions for FEA Pre and Post Processing*. Keynote Speech 2nd Conference on Advances and Applications of GiD. Barcelona, Spain.

[2]   Pace, M. E. 1994. *Computer-Aided Structural Engineering (CASE) Project (User's Guide)*, Vicksburg, MS: U.S. Army Engineer Waterways Experiment Station.

[3]   Sacks, R., C. M. Eastman, and G. Lee, "Parametric 3D Modeling in Building Construction with Examples from Precast Concrete", *Automation in Construction*, Vol. 13, No. 3, 2003, pp. 291-312.

[4]   Dawkins, W. P. 1994. *Computer Program for Two-Dimensional Dynamic Analysis of U-Frame or W-Frame Structures (CDWFRM).* Vicksburg, MS: U. S. Army Engineer Waterways Experiment Station.

[5]   Riveros, G. A. and V. S. Gopalaratnam. 2009. *A Non-Linear Fracture Mechanics Numerical Solution for Reinforced Concrete Deep*

Beams: A Theoretical Manual for the Fracture Mechanics Analysis of Reinforced Concrete Beams (FMARCB) Program, ERDC/ITL TR-09-5. Vicksburg, MS: U.S. Army Engineer Research and Development Center.

[6]   Riveros G. A. and Gopalaratnam V. S.," Shear Response of Reinforced Concrete Deep Beam: Validating Fracture Mechanics Based Numerical Modelling With Experiments," ACI, SP-300 Fracture Mechanics Applications in Concrete, 2015.

[7]   Riveros, G. A. and V. S. Gopalaratnam. 2008. U*ser's Guide: Fracture Mechanics Analysis of Reinforced Concrete Beams (FMARCB)*, ERDC/ITL TR-08-1. Vicksburg, MS: U.S. Army Engineer Research and Development Center.

[8]   Riveros, G. A. and V. S. Gopalaratnam. 2015. *Shear Response of Reinforced Concrete Deep Beam: Validating Fracture Mechanics Based Numerical Modelling With Experiments*, ACI S P 300 on Fracture Mechanics. .

[9]   International Alliance for Interoperability. 2004. *Industry Foundation Classes, IFC2x Edition 2 Addendum 1.*

[10] Weaver, W. Jr. and P. R. Johnston. 1984. *Finite Elements for Structural Analysis*, Englewood Cliffs, NY: Prentice-Hall.

[11] Bathe, K. J. 2007. *Finite Element Procedures*, Englewood Cliffs, NY: Prentice-Hall.